# redAnTS 2 - Generate Finite-Element Model

Author: Rajesh Bhaskaran, Cornell University

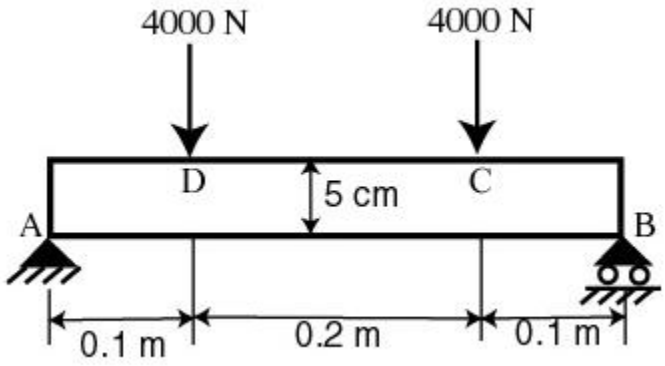


## Generate Finite-Element Model

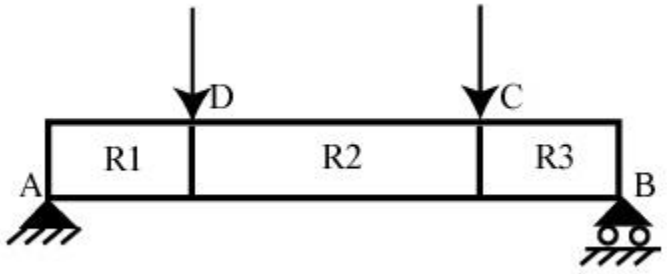
In the introductory tutorial, we generated the mesh using the Mapping Mesh Tool. Here we'll use the CSG Mesh Tool to create the geometry and mesh it. The latter is more convenient to use especially for complex geometries while the former gives you more control over the mesh.

### Strategy for Geometry Creation

We need to apply point boundary conditions at four points *A*, *B*, *C* and *D* shown in the figure below. In *redAnTS*, point boundary conditions are applied to nodes. When we mesh the rectangle using the CSG Mesh Tool, nodes will automatically be created at *A* and *B* since these are corner points; the corresponding displacement BC's can be applied to these corner nodes. However, there is no guarantee that there will be nodes exactly at *C* and *D* since these are not corners. In this case, *redAnTS* will apply the forces at the nodes that are closest to *C* and *D*. This is possibly acceptable if the mesh is sufficiently fine.




An alternative scenario is that we get clever and force *C* and *D* to be corner points. This can be done by dividing the geometry into three rectangles *R1*, *R2* and *R3*as shown below. This will force nodes to be created at points C and D. Then, the point forces can be applied to these nodes. This is the strategy we'll use in creating the mesh.




### Create Geometry

Under Mesh, click on Create CSGtool. This brings up the CSG Mesh Tool GUI.

Let's modify the drawing palette to help with the geometry creation. Turn on grid: **Options > Grid**

Enable snap to grid so that points are created exactly at grid points: **Options > Snap**

Change axes limits: **Options > Axes Limits**
X-axis range: `[-0.05 0.45]`
Y-axis range: `[-0.05 0.1]`
Click **Apply** and then **Close**.

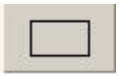
Increase number of grid lines: **Options > Grid Spacing ...**
Unselect Auto for both *x* and *y* linear spacing.
Change X-axis linear spacing to 0.025 while retaining the axis limits: `-0.05:0.025:0.45`
Similarly, change Y-axis linear spacing to 0.01.
Click **Apply** and then **Done**.

Enter *Draw* mode: **Draw > Draw Mode**

Create rectangle *R1*: **Draw > Rectangle/Square**
Click approximately near (0,0) and (0.1,0.05). Don't worry about getting these points exactly right since we will edit the coordinate values as below:
Double-click on *R1*
Left: 0
Bottom: 0
Width: 0.1
Height: 0.05
**OK**

Create rectangle *R2*: Note the row of icons below the menu options at the top. This is referred to as the toolbar. The icons in the toolbar offer shortcuts to options in the menus.

For example, the rectangle icon



offers a shortcut for creating a rectangle and is equivalent to the menu option we used for creating *R1*. Click on the rectangle icon. Click approximately near (0.1,0) and (0.3,0.05) to create *R2*. Modify coordinates:
Double-click on *R2*
Left: 0.1
Bottom: 0
Width: 0.2
Height: 0.05
**OK**

Look below the toolbar. You should see Set formula: R1+R2
You can go in and edit this formula. This facility allows you to arbitrarily add or subtract geometric components as necessary to create the final geometry.

Save: **File > Save Geometry As**
Filename: `beam.m`
This file can be read back into CSG Mesh Tool using **File > Open**

Create rectangle *R3* by copying *R1*:
Select *R1* by clicking once on it.
Copy R1: **Edit > Copy**
Paste R1: **Edit > Paste ...**
X-axis displacement: `0.3`
**OK**

You should see that R3 has been added to Set formula.


## Check Boundaries

Click on the boundary icon:

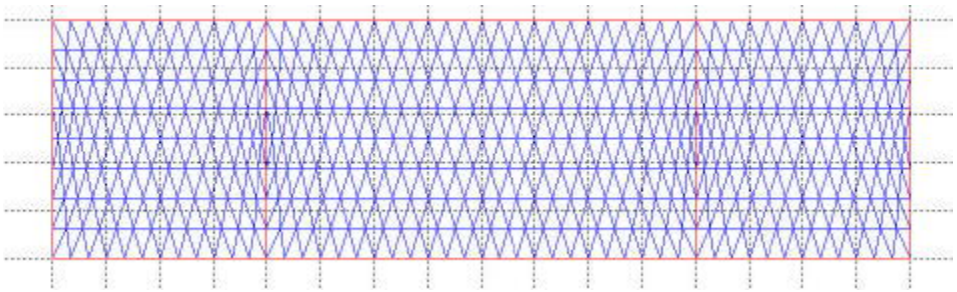

This shows the boundaries of the geometry. The external boundaries are shown in red and internal boundaries in black. In the next step, we'll specify boundary conditions for the external boundaries.

## Mesh Geometry

To initialize the mesh, click on the triangle icon:

To refine the mesh 2 times, select the multiple triangle icon *twice*:

Improve mesh quality: **Mesh > Jiggle Mesh**



Are there nodes at the points where the boundary conditions need to be applied?

Save geometry: **File > Save Geometry**

Note that the manual for the CSG Tool can be accessed by selecting the Help button in the GUI.

## Export Mesh

Export mesh into *redAnTS* GUI: **Mesh Done** (lower right)
Enter mesh filename: `beam_mesh.mat` (this file can be read from the redAnTS main GUI)
**OK**

This puts you back in the main *redAnTS* GUI. We see under Current Settings that the beam_mesh.dat file is loaded. To plot the mesh in this GUI, click on the drop-down menu under Plotting and select Mesh.

Let's move on to Step 3 where we will specify inputs such as material properties and boundary conditions.

**Go to Step 3: Specify Inputs**

Go to all MATLAB Learning Modules